Introduction to Heat Transfer Module

VERSION 4.3

COMSOL

Introduction to the Heat Transfer Module

Version: May 2012 COMSOL 4.3

Contact Information

Visit www.comsol.com/contact for a searchable list of all COMSOL offices and local representatives. From this web page, search the contacts and find a local sales representative, go to other COMSOL websites, request information and pricing, submit technical support queries, subscribe to the monthly eNews email newsletter, and much more.

If you need to contact Technical Support, an online request form is located at www.comsol.com/support/contact.

Other useful links include:

- Technical Support www.comsol.com/support
- Software updates: www.comsol.com/support/updates
- Online community: www.comsol.com/community
- Events, conferences, and training: www.comsol.com/events
- Tutorials: www.comsol.com/products/tutorials
- Knowledge Base: www.comsol.com/support/knowledgebase

Part No. CM020804

Contents

Introduction

The Heat Transfer Module is used by product designers, developers, and scientists, who use detailed geometrical descriptions to study the influence of heating and cooling in devices or processes. It has modeling tools for the simulation of all mechanisms of heat transfer including conduction, convection, and radiation. Simulations can be run for transient and steady conditions in 1D, 1D axisymmetric, 2D, 2D axisymmetric, and 3D space coordinate systems.

The high level detail provided by these simulations allows for the optimization of design and operational conditions in devices and processes influenced by heat transfer.

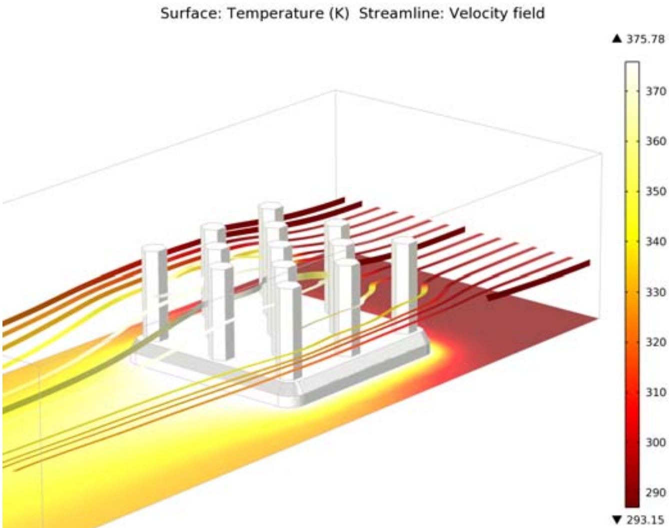


Figure 1: Temperature and flow field in an aluminum heat sink and in cooling air that is pumped over the heat sink. The temperature and flow field are solved using detailed geometry and a description of the physics.

The Heat Transfer Module's Model Library is a resource with examples of the heat transfer interfaces and the many available features. The Model Library has educational tutorials as well as industrial equipment and device benchmark models for verification and validation.

This introduction fine tunes your COMSOL model building skills for heat transfer simulations. The model tutorial solves a conjugate heat transfer problem from the field of electronic cooling but the principles can be applied to any other field involving heat transfer in solids and fluids.

Basic Concepts Described in The Heat Transfer Module

Heat is one form of energy that, similar to work, is in transit from inside a system or from one system to another. This energy may be stored as kinetic or potential energy in the atoms and molecules in a system.

Conduction is the form of heat transfer that can be described as proportional to the temperature gradients in a system. This is formulated mathematically by Fourier's law. The Heat Transfer Module describes conduction in systems with constant thermal conductivity and in systems where thermal conductivity is a function of temperature itself or of any other model variable, for example chemical composition.

In the case of a moving fluid, the energy transported by the fluid has to be modeled in combination with fluid flow. This is referred to as convection of heat and has to be accounted for in forced and free convection (conduction and convection). This module includes descriptions for heat transfer in fluids and conjugate heat transfer (heat transfer in solids and fluids in the one system) for both laminar and turbulent flows. In the case of turbulent flow the module offers high-Reynolds or, alternatively, low-Reynolds models to accurately describe conjugate heat transfer.

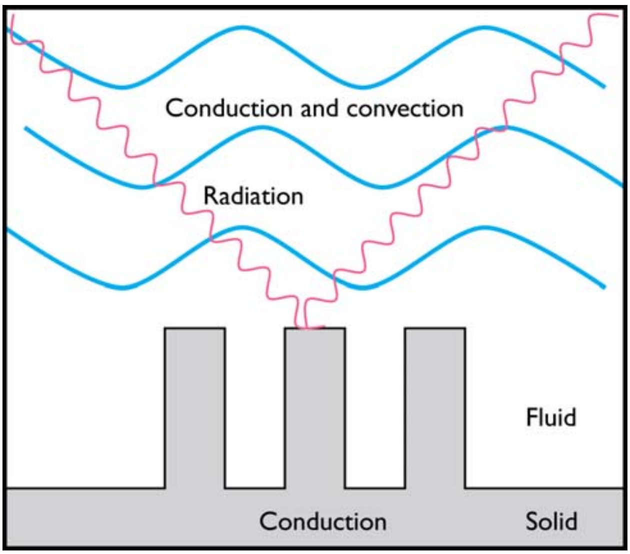


Figure 2: Heat transfer in a system containing a solid and a fluid (conjugate heat transfer). In the fluid, heat transfer can take place through conduction and convection, while conduction is the main heat transfer mechanism in solids. Heat transfer by radiation can occur between surfaces in a model.

Radiation is the third mechanism for heat transfer included in the module. It is modeled using expressions for surface-to-ambient radiation (for example, by defining boundary conditions) and also by using surface-to-surface radiation models, which includes external radiation sources (for example, the sun). The surface-to-surface radiation capabilities are based on the radiosity method. In addition, the module also contains functionality for radiation in participating media. This radiation model accounts for the absorption, emission, and scattering of radiation by the fluid present between radiating surfaces.

The basis of the Heat Transfer Module is the balance of energy in a studied system. The contributions to this energy balance originate from conduction, convection, and radiation but also from latent heat, Joule heating, heat sources, and heat sinks. In the case of moving solids, translational terms may also be included in the heat transfer models, for example for solids in rotating machinery. Physical properties and heat sources (or sinks) can be described as arbitrary expressions of the dependent variables in a model (for example, temperature and electric field). The heat transfer equations are defined automatically by the dedicated physics interfaces for heat transfer and fluid flow. The formulations of these equations can be visualized in detail for validation and verification purposes.

Physical properties such as thermal conductivity, heat capacity, density, and emissivity can be obtained from the built-in material library for solids and fluids and from the add-on Material Library in COMSOL. In addition, the module contains relations for the calculation of heat transfer coefficients for different types of convective heat transfer from a surface. For turbulent heat transfer, it also features relations that calculate the thermal conductivity in turbulent flow using the eddy diffusivity from turbulence models (sometimes referred to as turbulent conductivity).

The work flow in the module is straightforward and is described by these steps—define the geometry, select the material to be modeled, select the type of heat transfer, define the boundary and initial conditions, define the finite element mesh, select a solver, and visualize the results. All these steps are accessed from the COMSOL Desktop. The mesh and solver steps are often automatically completed with the default settings, which are already tuned for each type of heat transfer interface.

The Applications

Heat transfer is present in most physical processes and phenomena, either as a side effect or as a desired effect. The module can be effectively used to study a variety of processes, for example to include building ventilation effects, to account for turbulent free convection and heat transfer, and to analyze the impact of electronic microdevice heat generation and cooling.

The Heat Transfer Module's Model Library contains tutorial and benchmark models from different engineering fields and applications. See "Opening the Model Library" to find out how to access the models.

The Electronics and Power Systems section in the Model Library includes models that often involve heat generation and heat transfer in solids and conjugate heat transfer, where cooling is described in greater detail. The models in these applications are often used to design cooling systems and control the operating conditions of electronic devices and power systems. When the model results are interpreted, it provides the tools needed to understand and optimize the flow and heat transfer mechanisms in these systems.

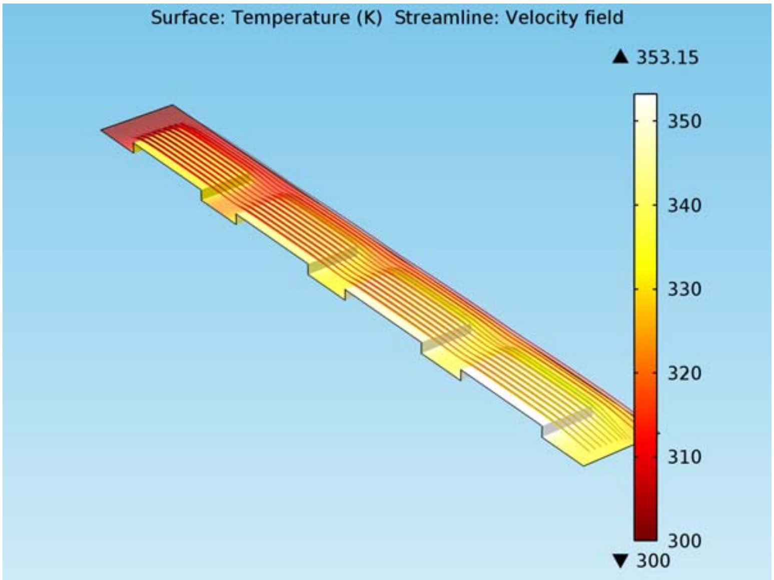


Figure 3: Temperature field as a result of conjugate heat transfer in a benchmark model for electronics cooling. This plot is from the model Convection Cooling of Circuit Boards—3D Forced Convection.

The Process and Manufacturing section in the Model Library has examples including thermal processing of materials and phase change processes such as continuous casting and friction stir welding. The common feature with these models is that the temperature field and the temperature variations have a very large impact on the material properties of the modeled processes and devices. The highly nonlinear behavior of the material properties as a function of temperature, combined with the high temperatures that implies accounting for radiation, make these processes difficult to predict and understand. Modeling and simulation often provide a shortcut to this understanding, which is required to design and operate a system.

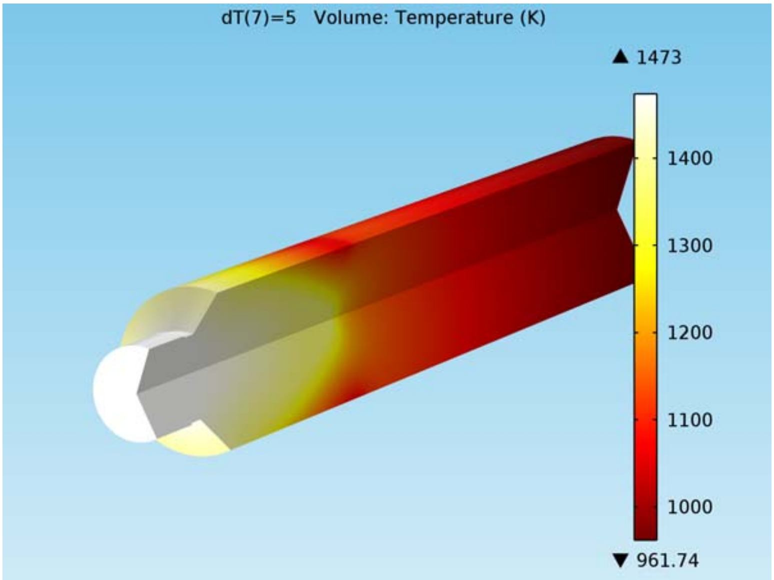


Figure 4: Temperature field plot from the Continuous Casting model. A sharp temperature gradient is found across the mushy layer, where the liquid metal solidifies.

The Medical Technology section in the Model Library introduces the concept of bioheating, where the influence of various processes in living tissue are accounted for as contributing to heat flux and as sources and sinks in the heat balance. The types of bioheating applications modeled include hyperthermia cancer therapy, such as microwave heating of tumors, and the interaction between microwave antennas and living tissue, for example, the influence of cellphone use to the temperature of tissue close to the ear. The benefit of using the bioheat equation is that it has been validated for different types of living tissue using empirical data for the different properties, sources, and sinks. The models and simulations defined in this interface provide excellent complements to experimental and clinical trials, which may be used, for example, to develop new methods for dose planning.

Opening the Model Library

To open a Heat Transfer Module Model Library model, select **View > Model Library** from the main menu in COMSOL Multiphysics. In the Model Library window that opens, expand the Heat Transfer Module folder and browse or search the contents. Click **Open Model and PDF** to open the model in COMSOL Multiphysics and a PDF to read background theory about the model including the step-by-step instructions to build it.

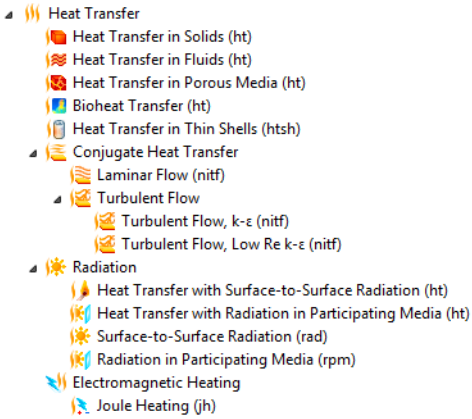
The MPH-files in the COMSOL model libraries can have two formats—Full MPH-files or Compact MPH-files.

- Full MPH-files, including all meshes and solutions. In the Model Library these models appear with the icon. If the MPH-file's size exceeds 25MB, a tip with the text "Large file" and the file size appears when you position the cursor at the model's node in the Model Library tree.
- Compact MPH-files with all settings for the model but without built meshes and solution data to save space on the DVD (a few MPH-files have no solutions for other reasons). You can open these models to study the settings and to mesh and re-solve the models. It is also possible to download the full versions—with meshes and solutions—of most of these models through Model Library Update. In the Model Library these models appear with the icon. If you position the cursor at a compact model in the Model Library window, a No solutions stored message appears. If a full MPH-file is available for download, the corresponding node's context menu includes a Model Library Update item.

A model from the Model Library is used as a tutorial in this guide. See "Tutorial Example—Heat Sink" starting on page 15. The next section describes the available interfaces in this module.

The Heat Transfer Module Interfaces

The figure below shows the Heat Transfer interfaces included in the Heat Transfer Module. These physics interfaces describe different heat transfer mechanisms and also include predefined expressions for sources and sinks. The Heat Transfer interfaces are available in 1D, 2D, 2D axisymmetric, 3D, and for stationary and time-dependent analyses.



Heat Transfer

The Heat Transfer in Solids interface ( ) describes, by default, heat transfer by conduction. It is also able to account for heat flux due to translation in solids, for example, the rotation of a disk or the linear translation of a shaft.

The Heat Transfer in Fluids interface ( ) accounts for conduction and convection in gases and liquids as the default heat transfer mechanisms. The coupling to the flow field in the convection term may be entered manually in the user interface or it may be selected from a list that couples heat transfer to an existing fluid flow interface. The Heat Transfer in Fluids interface may be used when the flow field has already been calculated and the heat transfer problem is added afterwards, typically for simulations of forced convection.

The Heat Transfer in Porous Media interface ( ) combines conduction in a porous matrix and in the fluid contained in the pore structure with the convection of heat generated by the flow of the fluid. This physics interface provides a power law or user-defined expression for the effective heat transfer properties and a predefined expression for dispersion in porous media. Dispersion is caused by the tortuous path of the liquid in the porous media, which would not be described if only the mean convective term was taken into account. This interface may be used for a wide range of porous materials, from porous structures in the pulp and paper industry to the simulation of heat transfer in soil and rock.

The Bioheat Transfer interface ( ) is a dedicated interface for heat transfer in living tissue. The bioheat model described in this interface has been verified for different types of living tissue, where also empirical data is available for physical properties, sources, and sinks. Apart from data such as thermal conductivity, heat capacity, and density, tabulated data is also available for blood perfusion rates and metabolic heat sources.

The Heat Transfer in Shells interface ( ) contains descriptions for heat transfer where large temperature variations may be present in a 3D structure but where the temperature differences across the thickness of the material of the structure are negligible. Typical examples may be structures like tanks, pipes, heat exchangers, airplane fuselages, and so forth. This physics interface can be combined with other heat transfer interfaces, for example to model the walls of a tank using the Heat Transfer in Thin Shells interface while the Heat Transfer in Fluids feature may be used to model the fluid inside the tank. In many cases using the Highly Conductive Layer boundary condition is the easiest solution.

Conjugate Heat Transfer

The Conjugate Heat Transfer interfaces ( ) describe heat transfer in solids and fluids and nonisothermal flow in the fluid. The heat transfer process is tightly coupled with the fluid flow problem and the physics interfaces include features for describing heat transfer in free and forced convection, including pressure work and viscous heating. These physics interfaces are available for laminar and turbulent nonisothermal flow. For highly accurate simulations of heat transfer between a solid and a fluid in the turbulent flow regime, low-Reynolds turbulence models resolve the temperature field in the fluid all the way to the solid wall in the Turbulent Flow, Low-Re k-ε interface ( ). The standard k-ε turbulence model in the Turbulent Flow, k-ε interface ( ) is computationally inexpensive compared to the other turbulent models but also usually less accurate.

If you also have the CFD Module, two additional turbulent models are available. For less computationally expensive simulations, where wall heat transfer is less important, the k-ω turbulence model gives good accuracy at a comparably low cost. The Spalart-Allmaras interface is a dedicated interface for conjugate heat transfer in aerodynamics, for example for the simulation of wing profiles.

Radiation

The Heat Transfer interface for radiation belong to essentially two different groups of radiation modeling interfaces: the surface-to-surface radiation and the radiation in participating media interfaces. The Heat Transfer with Surface-to-Surface Radiation interface ( ) combines heat transfer in fluid or solids including conduction and convection with surface-to-surface radiation. The Heat Transfer with Radiation in Participating Media interface ( ) combines heat transfer by conduction and convection in solids and fluids with radiation where absorption or emission of radiation is taken into account by the radiation model. In addition the Surface-to-Surface Radiation interface ( ) describes systems where only radiation is computed, typically to estimate radiation between surfaces in space applications where the surface temperature is known. The corresponding Radiation in Participating Media interface ( ) computes the radiation, including absorption and emission effects, in a media where the temperature is known.

Electromagnetic Heating

The Joule Heating interface ( ) is a multiphysics interface that couples electric heating and current conduction in electric conductors with heat transfer for modeling of Joule heating (resistive heating). This multiphysics interface includes the features from the Heat Transfer in Solids interface with the Electric Current interface including predefined couplings for Joule heating.

Physics List by Space Dimension and Study Type

The table lists the physics interfaces available with this module in addition to those included with the COMSOL basic license.

PHYSICS	ICON	TAG	SPACE DIMENSION	PRESET STUDIES
Fluid Flow				
Single-Phase Flow				
Single-Phase Flow, Laminar Flow*		spf	3D, 2D, 2D axisymmetric	stationary; time dependent
Turbulent Flow, k-ε		spf	3D, 2D, 2D axisymmetric	stationary; time dependent
Turbulent Flow, Low Re k-ε		spf	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
Non-Isothermal Flow				
Laminar Flow		nitf	3D, 2D, 2D axisymmetric	stationary; time dependent
Turbulent Flow, k-ε		nitf	3D, 2D, 2D axisymmetric	stationary; time dependent

PHYSICS	ICON	TAG	SPACE DIMENSION	PRESET STUDIES
Turbulent Flow, Low Re k-ε		nitf	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
Heat Transfer				
Heat Transfer in Solids*		ht	all dimensions	stationary; time dependent
Heat Transfer in Fluids*		ht	all dimensions	stationary; time dependent
Heat Transfer in Porous Media		ht	all dimensions	stationary; time dependent
Bioheat Transfer		ht	all dimensions	stationary; time dependent
Heat Transfer in Thin Shells		htsh	3D	stationary; time dependent
Conjugate Heat Transfer				
Laminar Flow		nitf	3D, 2D, 2D axisymmetric	stationary; time dependent
Turbulent Flow, k-ε		nitf	3D, 2D, 2D axisymmetric	stationary; time dependent
Turbulent Flow, Low Re k-ε		nitf	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
Radiation				
Heat Transfer with Surface-to-Surface Radiation		ht	all dimensions	stationary; time dependent
Heat Transfer with Radiation in Participating Media		ht	3D, 2D	stationary; time dependent
Surface-to-Surface Radiation		rad	all dimensions	stationary; time dependent
Radiation in Participating Media		rpm	3D, 2D	stationary; time dependent
Electromagnetic Heating				
Joule Heating*		jh	all dimensions	stationary; time dependent

* This is an enhanced interface, which is included with the base COMSOL package but has added functionality for this module.

The next section starts the tutorial using the Heat Sink model from the Model Library.

Tutorial Example—Heat Sink

This model is an introduction to simulations of fluid flow and conjugate heat transfer. It demonstrates the following important points. How to:

- Draw an air box around a device in order to model convective cooling in the box.
- Set a total heat source on a domain using automatic volume computation.
- Model the temperature difference between two surfaces in the presence of a thin thermally resistive layer.
- Include the radiative heat transfer between surfaces in a model.
- Display results in an efficient way using selections in data sets.

The modeled system consists of an aluminum heat sink for the cooling of an electronic component. See Figure 5

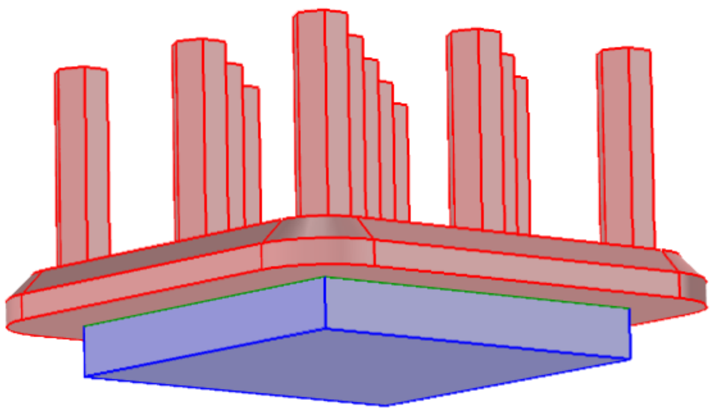


Figure 5: Heat sink and electronic component geometry

The heat sink is mounted inside a channel of rectangular cross section (see Figure 5). Such a set-up is used to measure the cooling capacity of heat sinks. Air enters the channel at the inlet and exits the channel at the outlet. To improve the thermal contact between the base surface of the heat sink and the top surface of the electronic component, thermal grease is used. All other external faces are thermally insulated. The heat dissipated by the electronic component is equal to 1W and is distributed in all the component volume.

The cooling capacity of the heat sink can be determined by monitoring the temperature in the electronic component.

The model solves a thermal balance for the electronic component, the heat sink, and the air flowing in the rectangular channel. Thermal energy is transported through conduction in the electronic component and the aluminum heat sink. The temperature field is discontinuous at the interface between the electronic component and the heat sink due to the presence of a thin resistive layer (thermal grease). Thermal energy is transported through conduction and convection in the cooling air. The temperature field is continuous across the internal surfaces between the heat sink and the air in the channel. The temperature is set at the inlet of the channel. A total power of 1 W is dissipated in the electronic component. The transport of thermal energy at the outlet is dominated by convection.

In the first step of the model, heat transfer by radiation between surfaces has been neglected. This assumption is valid as the surfaces have low emissivity (close to 0), which is usually the case for polished metals. When the surface emissivity is large (close to 1), the surface-to-surface radiation should then be considered. This is done in the second step of this tutorial. The model is modified to account for surface-to-surface radiation at the channel and heat sink boundaries. Assuming that the surfaces have been treated with black paint, the surface emissivity is close to 1 in this second case.

The flow field is obtained by solving one momentum balance for each space coordinate (x, y, and z) and a mass balance. The inlet velocity is defined by a parabolic velocity profile for fully developed laminar flow. At the outlet, a constant pressure is combined with the assumption that there are no viscous stresses in the direction perpendicular to the outlet. At all solid surfaces, the velocity is set to zero in all three spatial directions.

The thermal conductivity of air, the heat capacity of air, and the air density are all temperature-dependent material properties. You can find all of the settings mentioned in the physics interface for Conjugate Heat Transfer in COMSOL Multiphysics. You also find the material properties, including their temperature dependence, in the Material Browser.

Results

In Figure 6, the hot wake behind the heat sink is a sign of the convective cooling effects. The maximum temperature, reached in the electronic component, is slightly more than 382 K.

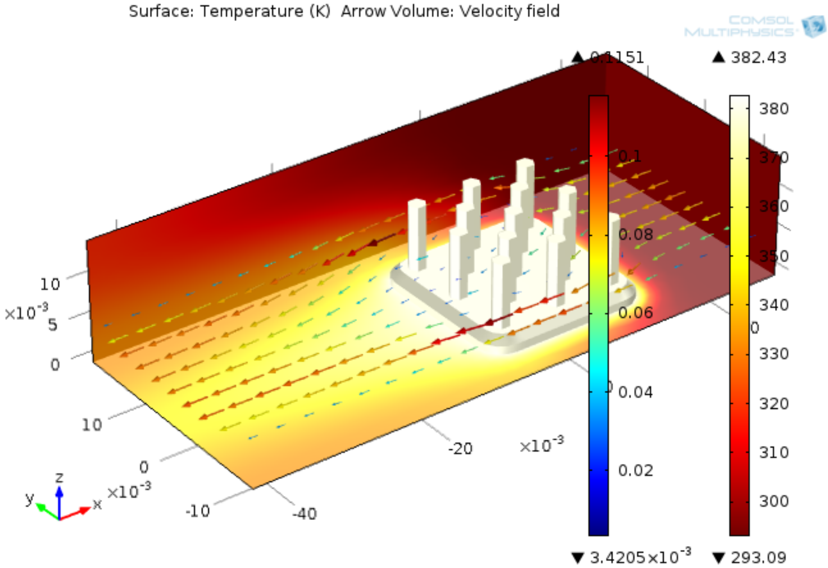


Figure 6: The surface plot shows the temperature field on the channel walls and the heat sink surface, while the arrow plot shows the flow velocity field around the heat sink.

In the second step, the temperature and velocity fields are obtained when surface-to-surface radiation is included and the surface emissivities are large. Figure 7 shows that the maximum temperature, slightly more than 360K, is decreased by about 22K compared to the first case in Figure 6. This confirms that radiative heat transfer is not negligible.

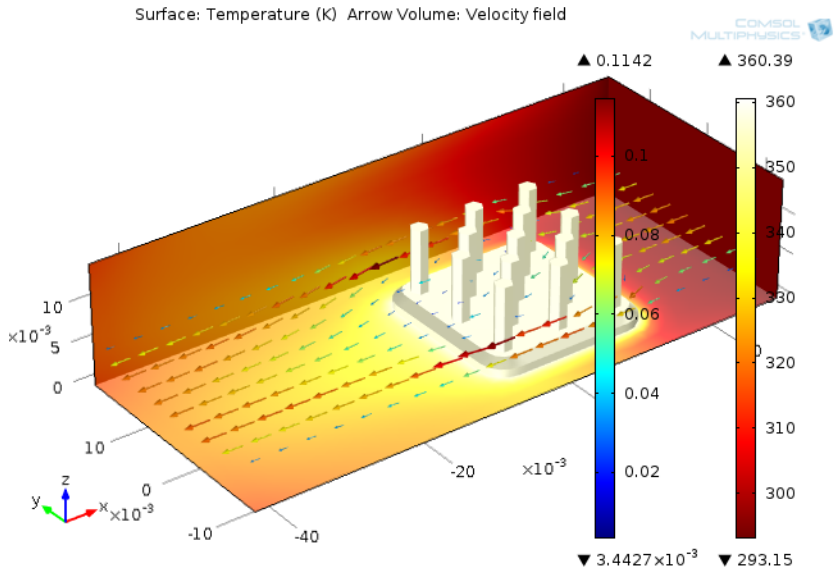


Figure 7: Effects of surface-to-surface radiation on temperature and velocity fields. The surface plot shows the temperature field on the channel walls and the heat sink surface, while the arrow plot shows the flow velocity field around the heat sink.

MODEL WIZARD

1. Open COMSOL Multiphysics. In the **Model Wizard** window the **Space Dimension** defaults to **3D**. Click **Next**.
2. In the **Add physics** tree under **Heat Transfer>Conjugate Heat Transfer**, double-click **Laminar Flow (nitf)** to add it to the **Selected physics** list. Click **Next**.
3. In the **Studies** tree, under **Preset Studies** select **Stationary**.
4. Click **Finish**.

GLOBAL DEFINITIONS - PARAMETERS

In this section, some parameters are defined that are used to specify the channel dimensions.

1 In the **Model Builder**, right-click **Global Definitions** and choose **Parameters** .
2 Go to the **Parameters** settings window. In the **Parameters** table enter the following settings as in the figure.

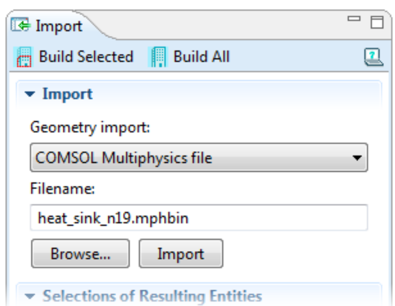
Parameters

Name	Expression	Value	Description
L_channel	7[cm]	0.070000 m	Channel length
W_channel	3[cm]	0.030000 m	Channel width
H_channel	1.5[cm]	0.015000 m	Channel height
L_chip	1.5[cm]	0.015000 m	Chip size
H_chip	2[mm]	0.0020000 m	Chip height
U0	5[cm/s]	0.050000 m/s	Mean inlet velocity
T0	20[degC]	293.15 K	Inlet temperature
P_tot	1[W]	1.0000 W	Total power dissipated by the electronic package

GEOMETRY 1

The geometry is built in three steps. First, the heat sink geometry is imported from a file. Then a block representing the chip is defined below the heat sink. Finally a volume representing the air channel is added.

Import 1



1 In the **Model Builder**, right-click **Geometry 1** and choose **Import** .
2 Go to the **Import** settings window and click **Browse**.
3 Navigate to your COMSOL Multiphysics installation folder, locate the subfolder **Heat_Transfer_Module\Tutorial_Models**, and select `heat_sink_n19.mphbin`

Note: The exact location of the files used in this exercise vary based on the installation. For example, if the installation is on your hard drive, the file path might be similar to `C:\Program Files\COMSOL43\models\`

4 Click **Import**.

Now define a work plane containing the bottom surface of the heat sink to draw the imprints of the chip and of the air channel.

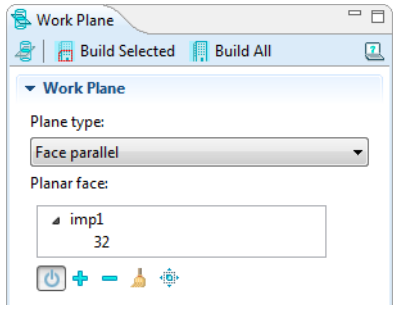
Work Plane 1

1 In the **Model Builder**, right-click **Geometry 1** and choose **Work Plane**.

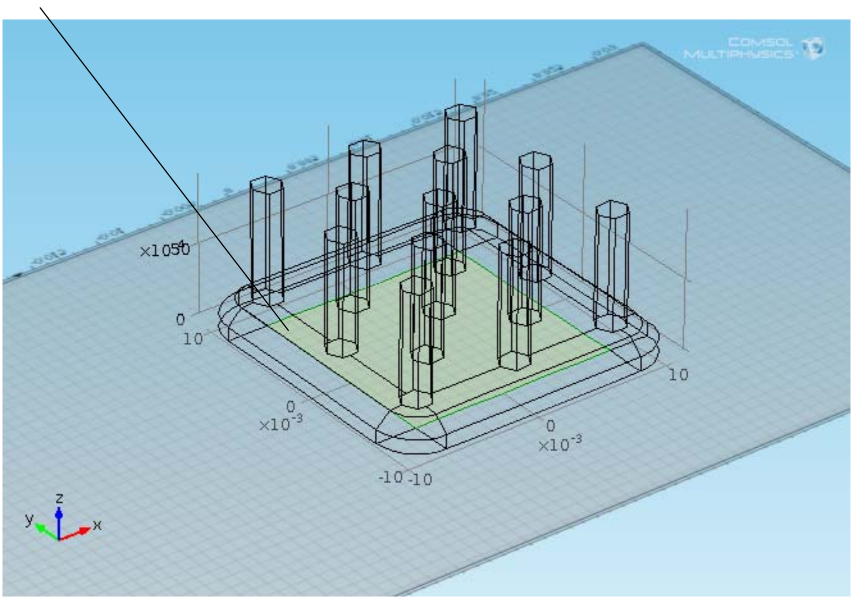
2 In the **Work Plane** settings window, locate the **Work Plane** section.

3 From the **Plane type** list, choose **Face parallel**.

4 On the object **imp1**, select Boundary 32 that corresponds to the face at the centre of the bottom side of the heat sink.

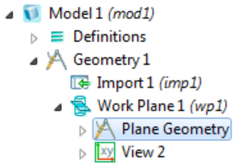


Boundary 32

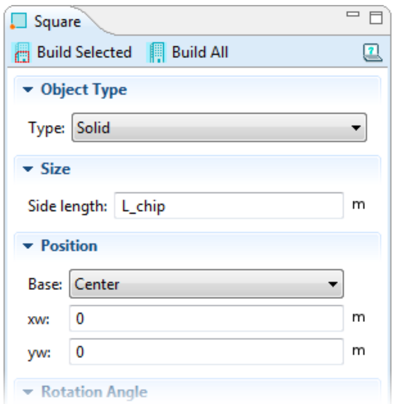


Square 1

1. In the **Model Builder**, under **Model 1>Geometry 1>Work Plane 1** right-click **Plane Geometry** and choose **Square**.

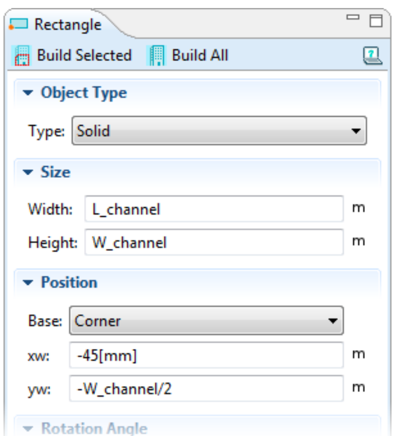


2. In the **Square** settings window, locate the **Size** section. In the **Side length** field, enter `L_chip`.
3. Under **Position** from the **Base** list, choose **Center**.
4. Click the **Zoom Extents** button on the **Graphics** toolbar.



Rectangle 1

1. In the **Model Builder**, right-click **Plane Geometry** and choose **Rectangle**.
2. In the **Rectangle** settings window, under **Size**:
   - In the **Width** field, enter `L_channel`.
   - In the **Height** field, enter `W_channel`.
3. In the **Rectangle** settings window under **Position**:
   - In the **xw** edit field, enter `-45[mm]`.
   - In the **yw** edit field, enter `-W_channel/2`.



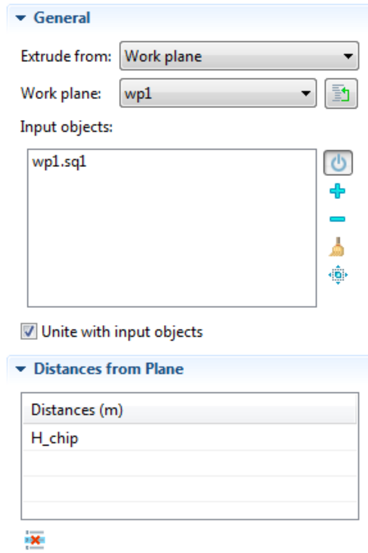
Now extrude the chip imprint to define the chip volume.

Extrude 1

1. In the **Model Builder**, right-click **Geometry 1** and choose **Extrude**.

2 Remove the object **wp1.r1** from the default selection so that only **wp1.sq1** remains in the **Input objects** list in the **Extrude** settings window under **General**.

3 In the **Extrude** settings window, under **Distances from Plane** enter `H_chip` in the table.



To finish the geometry extrude the channel imprint **wp1.r1** in the opposite direction to define the air volume.

Extrude 2

1 In the **Model Builder**, right-click **Geometry 1** and choose **Extrude**.

2 In the **Extrude** settings window, under **Distances from Plane** enter `H_channel` in the table

3 Select the **Reverse direction** check box.

In order to simplify the geometry and to improve the mesh that will be generated later, you now form composite faces. To facilitate face selection, use the wireframe rendering option.

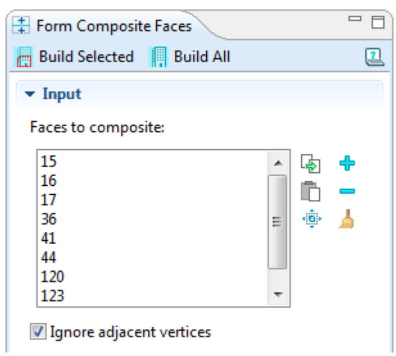
Form Composite Faces 1

1 On the **Graphics** toolbar, click the **Go to Default 3D View** button and then the **WireFrame Rendering** button.

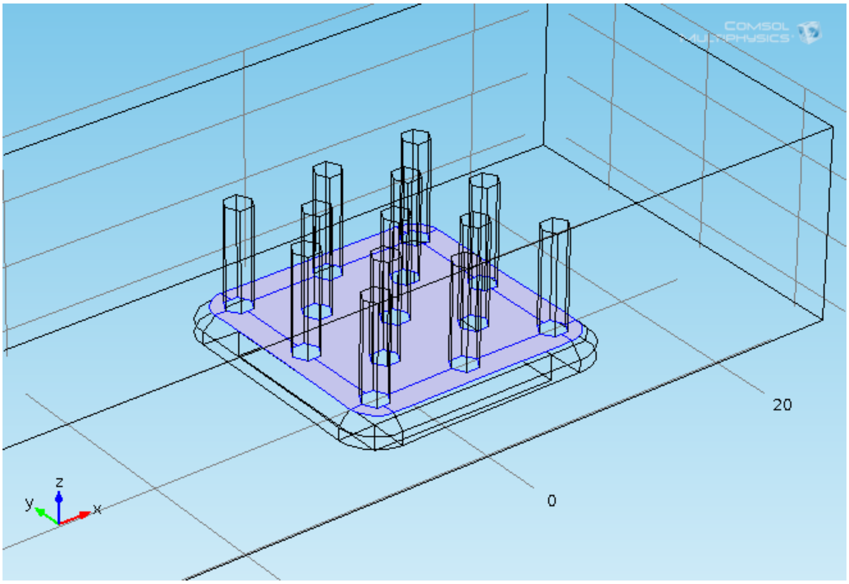
2 In the **Model Builder**, right-click **Geometry 1** and choose **Virtual Operations>Form Composite Faces**.

3 On the object **fin**, select Boundaries 15–17, 36, 41, 44, 120, 123, and 125 only.

Note: There are many ways to select geometric entities. When you know the domain to add, such as in this exercise, you can click the **Paste Selection** button and enter the information in the **Selection** field. For more information about selecting geometric entities in the **Graphics** window, see the *COMSOL Multiphysics User's Guide*.

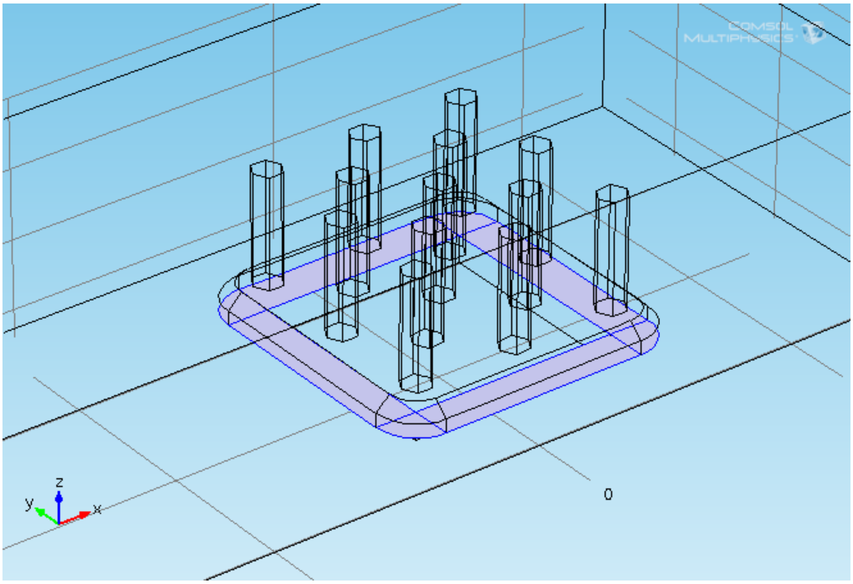


This selection is shown in this figure:

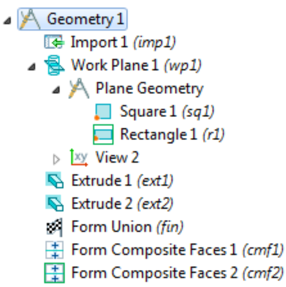


Form Composite Faces 2

1 In the **Model Builder**, right-click **Geometry 1** and choose **Virtual Operations>Form Composite Faces**.
2 In the object **cmf1**, select Boundaries 8, 9, 13, 32, 39, 113, 116, and 117 only. This selection is shown in this figure:



The final node sequence under **Geometry** should match this figure.

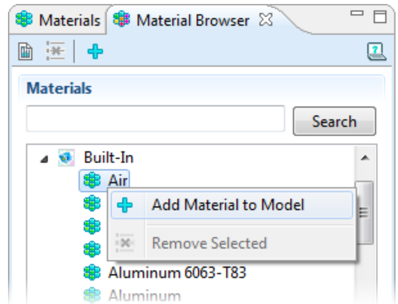


MATERIALS

Air and Aluminum 3003-H18

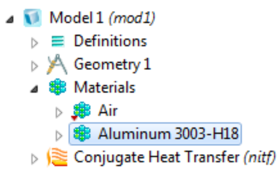
1 From the main menu, select **View>Material Browser**.

2. In the **Material Browser** window in the **Materials** tree under **Built-In**, right-click **Air** and choose **Add Material to Model**.



By default, the first material added applies to all domains. Typically, you can leave this setting and add other materials that override the default material where applicable. In this example, specify aluminum for Domain 2.

3. Go to the **Material Browser** window.
4. In the **Materials** tree under **Built-In**, right-click **Aluminum 3003-H18** and choose **Add Material to Model**.
5. In the **Model Builder**, click **Aluminum 3003-H18**.



6. In the **Graphics** window, select Domain 2 only and add it to the **Selection** list.

Silica Glass

1. Go to the **Material Browser**. In the **Materials** tree under **Built-In**, right-click **Silica glass** and choose **Add Material to Model**.
2. In the **Model Builder**, click **Silica glass**.
3. Select Domain 3 only.

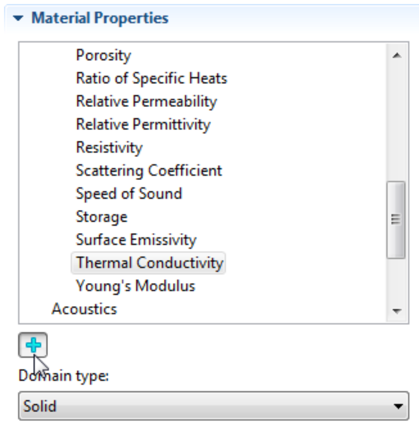
Thermal Grease

1. In the **Model Builder**, right-click **Materials** and choose **Material**.
2. In the **Material** settings window, locate the **Geometric Entity Selection** section.
3. From the **Geometric entity level** list, choose **Boundary**. Select Boundary 34 only.
4. Right-click **Material 4** and choose **Rename** (or press F2). Enter `Thermal Grease` in the **New name** field. Click **OK**.

5 In the **Material** settings window, click to expand the **Material Properties** section.

6 In the **Material properties** tree, select **Basic Properties>Thermal Conductivity**.

7 Click the **Add to Material** button under the table.

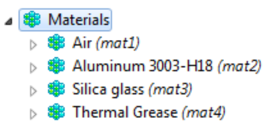


8 Under the **Material Contents** section, in the table, enter the following settings:

Material Contents

	Property	Name	Value	Unit	Property group
✓	Thermal conductivity	k	2[W/m/K]	W/(m*K)	Basic

The final node sequence under **Materials** should match this figure.



CONJUGATE HEAT TRANSFER (NITF)

Now define the physical properties of the model. Start by adding a Fluid feature to define the fluid domain.

Fluid 1

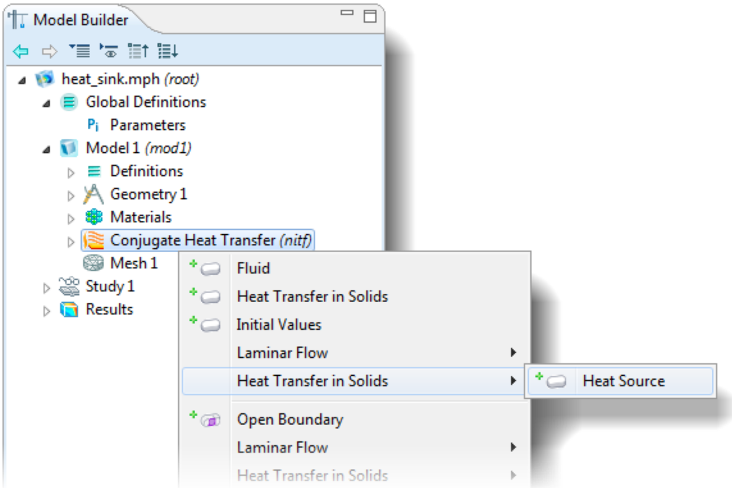
1 In the **Model Builder**, right-click **Conjugate Heat Transfer (nitf)** and choose **Fluid**.

2 In the **Fluid** settings window select Domain 1 only.

Next use the power parameter to define the total heat source equal in the electronic package.

Heat Source 1

1 In the **Model Builder**, right-click **Conjugate Heat Transfer** and choose the domain setting **Heat Transfer in Solids>Heat Source**.



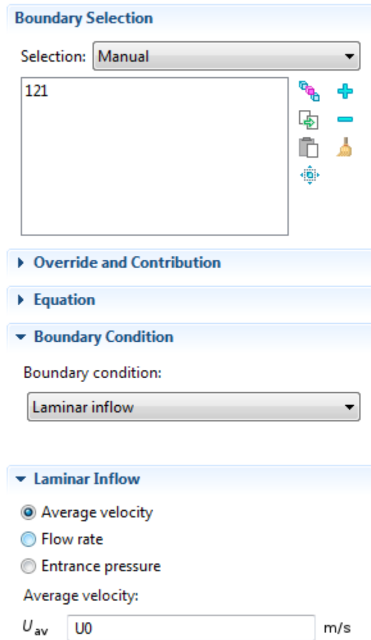
2 In the **Heat Source** settings window, under **Domain Selection**, select Domain 3 only.

3 Under the **Heat Source** section, click the **Total power** button. In the P_{tot} field, enter `P_tot`.

For the default **Wall** node, **No slip** is the default boundary condition for the fluid. Define the inlet and outlet conditions as described below.

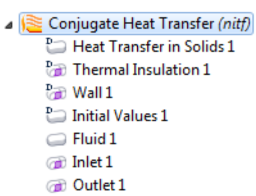
Inlet 1

1. In the **Model Builder**, right-click **Conjugate Heat Transfer (nitf)** and choose the boundary condition **Laminar Flow>Inlet**.
2. In the **Inlet** settings window, select Boundary 121 only.
3. Under **Boundary Condition** from the **Boundary condition** list, select **Laminar inflow**.
4. Under **Laminar Inflow** in the U_{av} field, enter U0.



Outlet 1

1. In the **Model Builder**, right-click **Conjugate Heat Transfer (nitf)** and choose the boundary condition **Laminar Flow>Outlet**.
2. Click the **Zoom Extents** button on the **Graphics** toolbar.
3. In the **Outlet** settings window select Boundary 1 only.

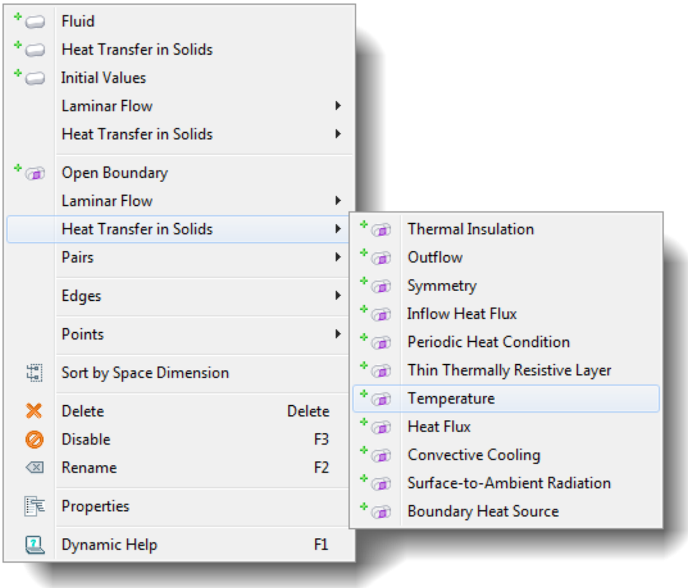


The node sequence in the **Model Builder** should match this figure so far. The 'D' in the upper left corner of a node means it is a default node.

Thermal insulation is the default boundary condition for the temperature. Define the inlet temperature and the outlet condition.

Temperature 1

1 In the **Model Builder**, right-click **Conjugate Heat Transfer (nitf)** and from the boundary level choose **Heat Transfer in Solids>Temperature** .



2 In the **Temperature** settings window select Boundary `121` only.

3 Under the **Temperature** section in the T_0 field, enter `T0`.

Outflow 1

1 In the **Model Builder**, right-click **Conjugate Heat Transfer (nitf)** and choose the boundary condition **Heat Transfer>Outflow** .

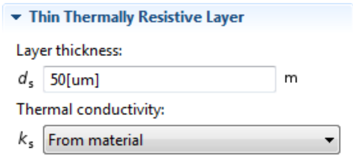
2 In the **Outflow** settings window select Boundary `1` only.

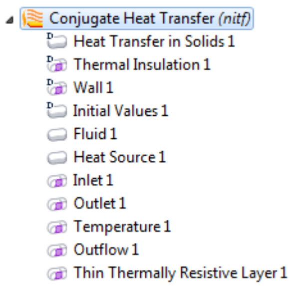
Thin Thermally Resistive Layer 1

1 In the **Model Builder**, right-click **Conjugate Heat Transfer (nitf)** and choose the boundary condition **Heat Transfer in Solids>Thin Thermally Resistive Layer** .

2 In the **Thin Thermally Resistive Layer** settings window select Boundary `34` only.

3 Under **Thin Thermally Resistive Layer** in the d_s field, enter `50[um]`.





The node sequence in the **Model Builder** under Conjugate Heat Transfer should match the figure.

MESH 1

Free Tetrahedral 1 and Size

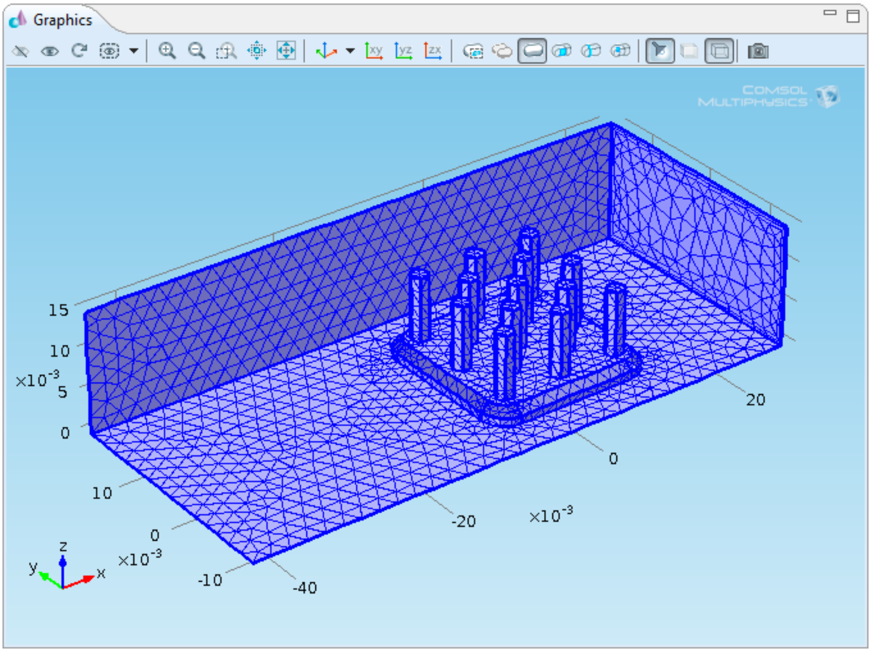
1 In the **Model Builder** under **Model 1 (mod1)**, click **Mesh 1**.

2 Go to the **Mesh** settings window. Under **Mesh Settings** from the **Element Size** list, select **Coarser**.

3 Click the **Build All** button.

To get a better view of the mesh, suppress some of the boundaries.

4 In the **Graphics** window, select Boundaries 1, 2, and 4 only. Click the **Hide Selected** button on the toolbar. The mesh is generated and displayed as shown in the figure below.



STUDY 1

1 In the **Model Builder**, right-click **Study 1** and choose **Compute**.

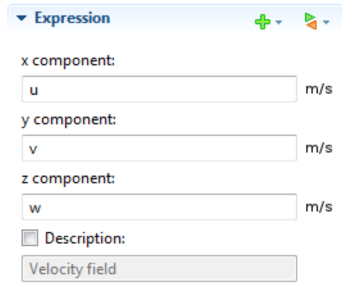
Two default plots are generated automatically. The first one shows the velocity magnitude on five parallel slices. The second one shows the temperature on the wall boundaries. Add an arrow plot to visualize the velocity field.

Temperature (nitf)

1 Under **Results** right-click **Temperature (nitf)** and choose **Arrow Volume**.

2 Go to the **Arrow Volume** settings window. Under **Data** from the **Data set** list, select **Solution 1**.

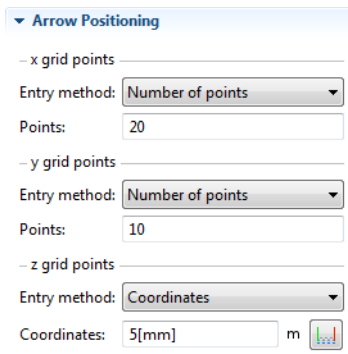
3 In the upper-right corner of the **Expression** section, click **Replace Expression**.



4 From the menu, choose **Conjugate Heat Transfer (Laminar Flow)>Velocity field (u,v,w)**. (Or enter `u`, `v`, and `w` in the fields).

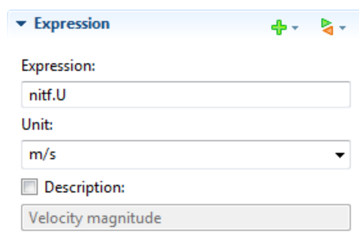
5 Under **Arrow Positioning**:

- In the **x grid points Points** field, enter `20`.
- In the **y grid points Points** field, enter `10`.
- In the **z grid points** subsection from the **Entry method** list, select **Coordinates**.
- In the **Coordinates** field, enter `5e-3` or `5[mm]`.



6 In the **Model Builder**, right-click **Arrow Volume 1** and choose **Color Expression**.

7 Go to the **Color Expression** settings window. In the upper-right corner of the **Expression** section, click **Replace Expression**.



8 From the menu, choose **Conjugate Heat Transfer (Laminar Flow)>Velocity magnitude (nitf.U)** (or enter `nitf.U` in the **Expression** field).

9 Click the **Plot** button.

The plot in Figure 6 on page 17 is displayed in the **Graphics** window.

Adding Surface-to-Surface Radiation Effects

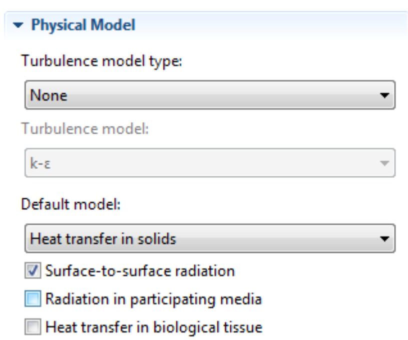
Now modify the model to include surface-to-surface radiation effects. First you need to enable surface to surface property in the physics interface.

In this second part we modify and solve again the model to study the effects of surface to surface radiation between the heat sink and the channel walls. Y

CONJUGATE HEAT TRANSFER

You can continue using the model built so far, or you can open the model from the Model Library.

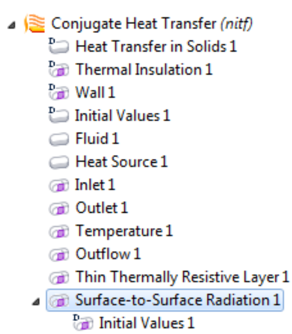
1. To open the model in the Model Library, select **View>Model Library**. Under the **Heat Transfer Module>Tutorial Models** folder, click **heat_sink** and click **Open**.
2. In the **Model Builder**, under **Model 1** click **Conjugate Heat Transfer**.
3. In the **Conjugate Heat Transfer** settings window, under the **Physical Model** section, click to select the **Surface-to-surface radiation** check box.



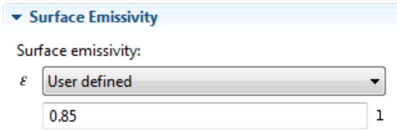
Now it is possible to add surface to surface boundary condition to the model.

Surface-to-Surface Radiation 1

1. Right-click **Conjugate Heat Transfer** and choose the boundary condition **Surface-to-Surface Radiation>Surface-to-Surface Radiation**. A **Surface-to-Surface Radiation** node with a second default **Initial Values** node is added to the **Model Builder**.
2. In the **Model Builder**, click the **Surface-to-Surface Radiation 1** node.
3. Select Boundaries 6,7,9–30,36–109, and 111–120 only.



4 In the **Surface-to-Surface Radiation** settings window, under **Surface Emissivity**, from the ε list, choose **User defined**. In the field, enter `0.85`.



By default the radiation direction is controlled by the opacity of the domains. Next define the opaque parts of the model.

Opaque 1

1 In the **Model Builder**, right-click the **Heat Transfer in Solids 1** node and choose **Opaque**.

Surface-to-Surface Radiation 2

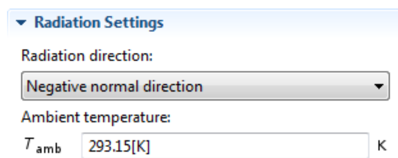
1 Add another **Surface-to-Surface Radiation** node. In the **Model Builder**, right-click **Conjugate Heat Transfer** and choose the boundary condition **Surface-to-Surface Radiation>Surface-to-Surface Radiation**.

2 Select Boundaries 2–5 only.

3 In the **Surface-to-Surface Radiation** settings window, under **Surface Emissivity** from the ε list, choose **User defined**. In the field, enter `0.9`.

4 Under **Radiation Settings**, from the **Radiation direction** list, choose **Negative normal direction**.

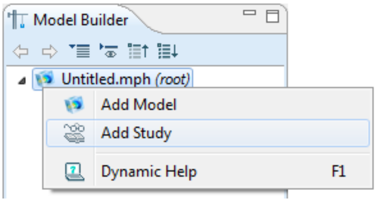
5 Highlight boundaries 1, 2, and 4. Click the **Hide Selected** button on the toolbar.



MODEL WIZARD

In order to keep the previous solution and to be able to compare it with this version of the model, add a second stationary study.

1 In the **Model Builder**, right-click the Root node and choose **Add Study**.



2 The **Add Study** window opens in the **Model Wizard**. Under **Preset Studies** select **Stationary**.

3 Click **Finish**.

STUDY 2

Step 1: Stationary

1 In the **Model Builder**, right-click **Study 2** and choose **Compute**.

RESULTS

The same default plots are generated as in "Results" on page 32. In this step, you edit the temperature plot to compare the results.

Temperature (nitf) 1

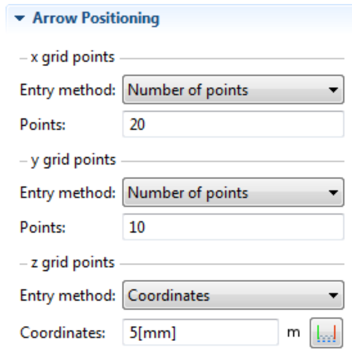
1 Under **Results** right-click **Temperature (nitf)** and choose **Arrow Volume**.

2 In the **Arrow Volume** settings window click the **Replace Expression** button.

3 From the menu, choose **Conjugate Heat Transfer (Laminar Flow)>Velocity field (u,v,w)**.

4 Under **Arrow Positioning**:

- In the **x grid points Points** field, enter `20`.
- In the **y grid points Points** field, enter `10`.
- In the **z grid points** subsection from the **Entry method** list, select **Coordinates**.
- In the **Coordinates** field, enter `5e-3` or `5[mm]`.



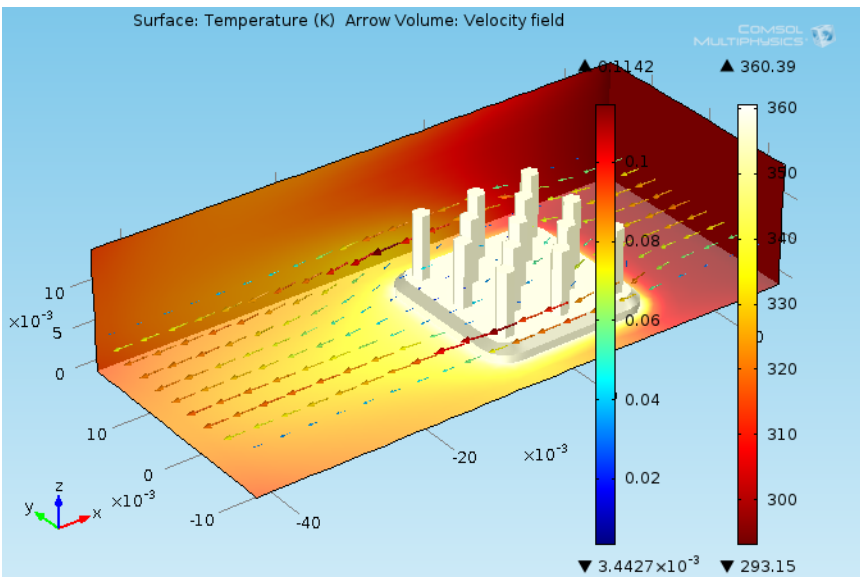
5 Under **Data** from the **Data set** list, choose **Solution 2**.

6 Under **Results**, right-click **Arrow Volume 1** and choose **Color Expression**.

7 In the **Color Expression** settings window click the **Replace Expression** button.

8 From the menu, choose **Conjugate Heat Transfer (Laminar Flow)>Velocity magnitude (nitf.U)**.

9 Click the **Plot** button.



As a final step, pick one of the plots to use as a model thumbnail.

1 In the **Model Builder** under **Results** click **Temperature (nitf)**.

2 From the **File** menu, choose **Save Model Thumbnail**.

To view the thumbnail image, click the **Root** node and look under the **Model Thumbnail** section. Make adjustments to the image in the **Graphics** window using the toolbar buttons until the image is one that is suitable to your purposes.